

MODELING OF HEAT TRANSFER AND ABLATION OF REFRACTORY MATERIAL DUE TO ROCKET PLUME IMPINGEMENT

Michael F. Harris

Sierra Lobo Inc. (Engineering Services Contract), Kennedy Space Center, FL

Bruce T. Vu

NASA, Kennedy Space Center, FL

ABSTRACT

C&R Technologies' Thermal Desktop and SINDA/FLUINT software were used in the thermal analysis of a flame deflector design for Launch Complex 39B at Kennedy Space Center (KSC), Florida. The analysis of the flame deflector took into account the heat transfer caused by plume impingement from the new rockets that are expected to be launched from KSC. The heat flux from the plume was computed using computational fluid dynamics (CFD) provided by Ames Research Center in Moffet Field, California. The results from the CFD solutions were mapped onto a 3-D Thermal Desktop model of the flame deflector, using the boundary condition mapping capabilities. The ablation subroutine in SINDA/FLUINT was then used to model the ablation of the refractory material.

INTRODUCTION

Kennedy Space Center is currently conducting an investigation of the launch-induced environment so that its facilities can meet the needs of the Space Launch System (SLS) and other space launch vehicles now in development. As part of this investigation, thermal analysis is being used to predict ablation. At present, the design of the flame deflector incorporates some form of refractory material, and it is necessary to predict loss of material as a result of rocket plume impingement. In the past, the analysis was performed using THERM1D, a one-dimensional ablation analysis software that limits the analysis to a specific location, as opposed to performing an analysis that covers the entire surface. Figure 1 shows an example result of the THERM1D analysis, indicating surface thickness with respect to time.

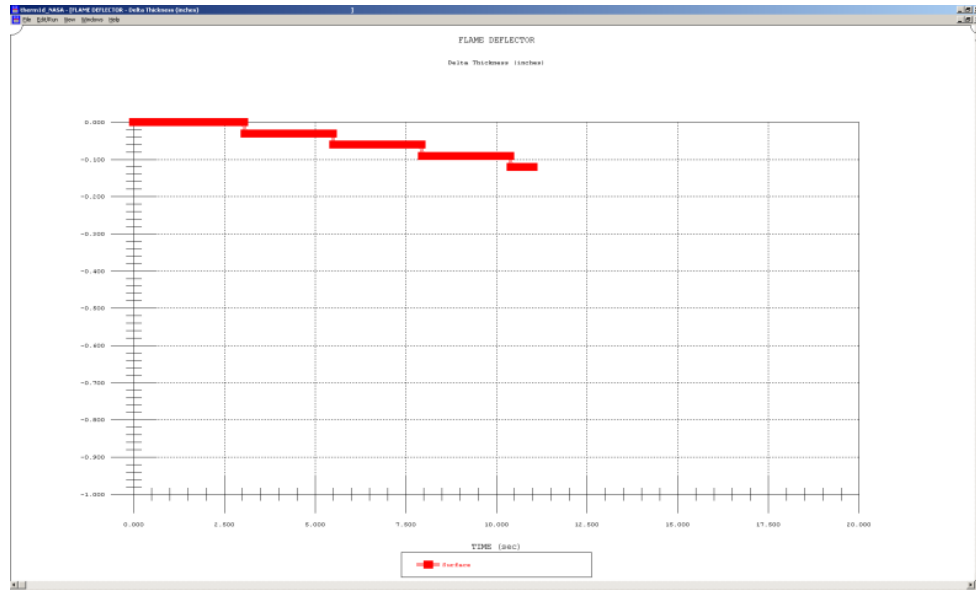


Figure 1. Example of THERM1D output.

Although the THERM1D software had proven to be sufficient in the past, by using C&R Technologies' Thermal Desktop software boundary condition mapper and ablation subroutine, the one-dimensional ablation analysis can be performed over the entire surface, providing the analyst with a contour plot of surface thickness and an animation. The boundary condition mapper allows for highly accurate CFD heat flux data that considers the dynamics of the transient high-velocity gas that impinges on the flame deflector to be mapped to the Thermal Desktop geometry. Once the data is mapped, Thermal Desktop will determine extent of the ablation over the flame deflector surface. (The current scope of this ablation analysis does not consider charring or pyrolysis of the material.) Figure 2 gives an example of the results obtained with Thermal Desktop.

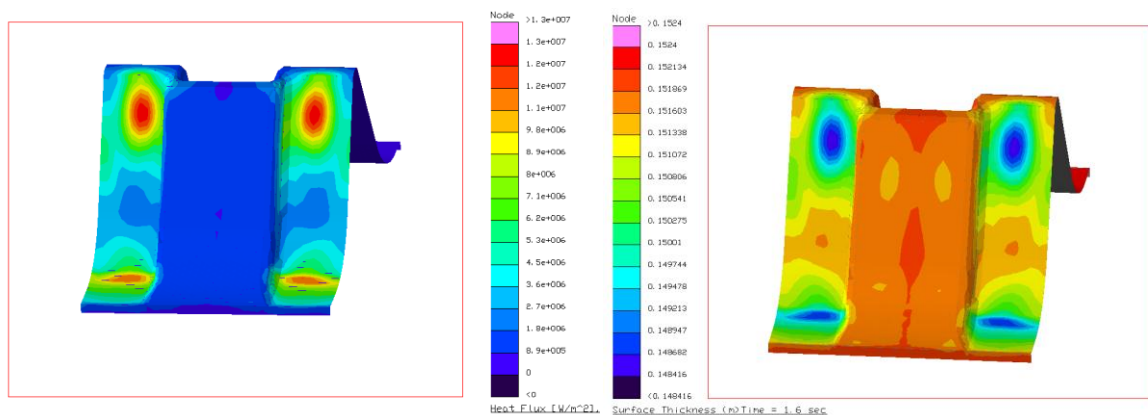


Figure 2. Thermal Desktop surface heat flux (L) and surface ablation thickness (R) examples.

Figure 3 shows some of the flame deflector concepts that underwent thermal analysis. The flame deflector used for the Space Transportation System (STS) is shown on the left, and the SLS concept is shown at the right.

MODEL SETUP: MESHING

The first step in the analysis is to create the model setup by importing the CAD geometry into NX/NASTRAN and, depending on the type of analysis, obtaining a surface mesh or solid mesh. Figure 3 shows the CAD geometry used for this analysis. The mesh is then imported into Thermal Desktop using the import features in the software, as shown in Figure 4.

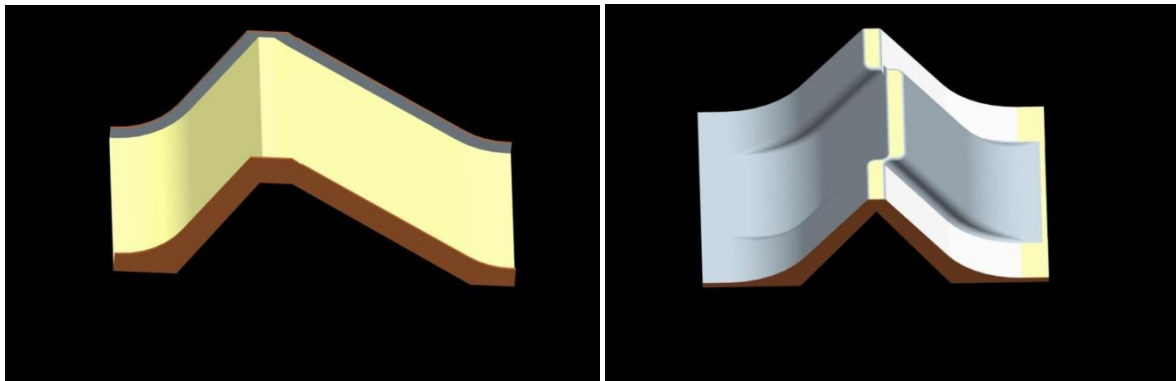


Figure 3. Flame deflector geometries.

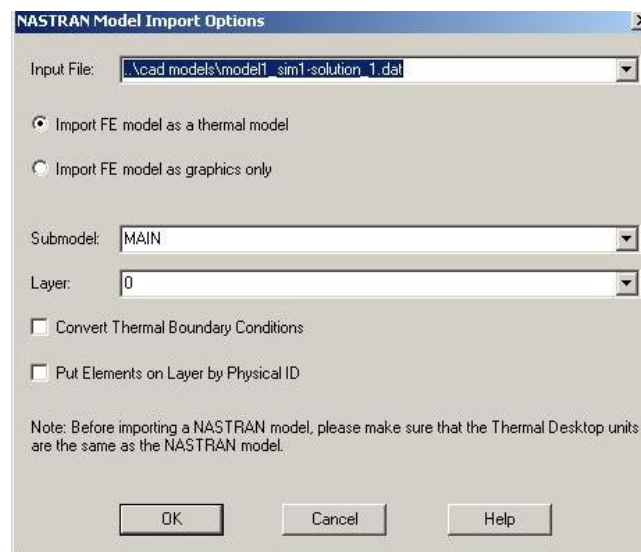


Figure 4. NASTRAN model import window.

Once the analyst imports the mesh, the thermal model will be displayed as an AutoCAD drawing. Figure 5 shows the Thermal Desktop models that were used for the flame deflector analysis. Either a 2-D surface mesh or a 3-D solid mesh can be used, depending on the analysis.

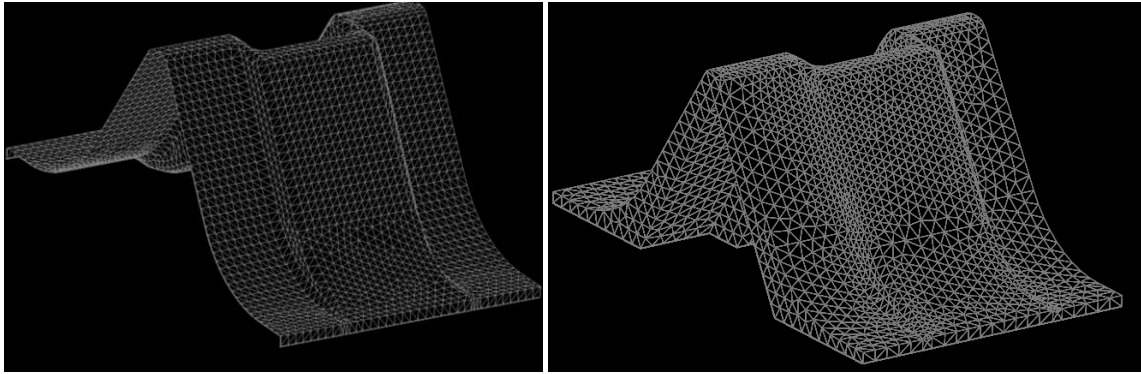


Figure 5. NASTRAN mesh imported into Thermal Desktop.

MODEL SETUP: DEFINING THERMOPHYSICAL PROPERTIES AND ABLATION NODES

After the refractory material is added and the properties of the material are defined, the ablation subroutine is selected from the Thermophysical Properties menu. Figure 6 shows the Thermophysical Properties menu for the refractory material. For this thermal analysis of the flame deflector, the refractory material was described as having an ablation temperature of 1373 K and the heat of ablation was 1.67 MJ/kg. The ablate routine was used for this analysis, but the ablate routine could be used in future analysis once experimental data is determined.

The screenshot shows the 'Thermophysical Properties' dialog box. The 'Property' is 'Refractory Material'. The 'Comment' field is empty. The 'Conductivity [W/m/K]' section has three rows for k, ky, and kz, each with a value of 0.504798, 0.999999, and 0.999999 respectively. The 'Specific Heat [J/kg/K]' section has a value of 1548.2. The 'Density [kg/m^3]' section has a value of 1842.12. The 'Effective emissivity' section has a value of 0. The 'Ablation' section is circled in red and contains the following information:

- ☒ Use Ablation
- Ablation Temp: 1373.15 K
- Heat of Ablation: 1.67473e+006 J/kg
- ☐ Allow complete burn through

Figure 6. Thermophysical Properties for Refractory Material

The ablation nodes then are specified by editing the Thin Shell Data menu for the surface elements. Under the Insulation tab, as shown in Figure 7, the analyst indicates that the

insulation is applied to either the top/outside or the bottom/inside surface. For this analysis, the top/outside surface was chosen. The material can be chosen from the drop-down menu, and a thickness can be specified. (The refractory material for the flame deflector was 6" thick.) The analyst must also specify the number of nodes for which the thickness must be discretized.

Figure 7. Defining ablation nodes on the surface.

BOUNDARY CONDITION MAPPER

The next step is to define the boundary conditions before executing the program and performing the analysis. The boundary condition for the flame deflector heat flux is computed by a transient conjugate heat transfer CFD code. The boundary condition mapper (BCM) feature of Thermal Desktop takes the transient surface heat flux data and maps it over the Thermal Desktop model surface.

To begin mapping, the analyst must first place the data in the appropriate format by defining the data type, heat flux or surface temperature, the units of the data, the coordinates of the nodes, and nodes that define the elements, specified as either triangles or quadrilaterals. For this analysis, a MATLAB script was developed in order to quickly format the CFD data, usually provided by ARC in a Tecplot format, into the required boundary condition mapper format to be read by Thermal Desktop. An example of the boundary condition mapper file format, taken from the Thermal Desktop User's Manual, is shown in Figure 8.

TEMPERATURE DEPENDENT HEAT FLUX BCM Sample Input

Note: Please note that all information including and after the '!' is for description and should not be in the actual file.

```
DATA: TEMPERATURE DEPENDENT HEAT FLUX
UNITS LENGTH meters
UNITS TEMPERATURE R
UNITS TIME SECONDS
UNITS DATA W/cm2
TEMPERATURES 2
300.000000
1000.000000
NODE 1 0. 0. 0.
NODE 2 0. 1. 0.
NODE 3 1. 0. 0.
NODE 4 1. 1. 0.
NODE 5 2. 0. 0.
NODE 6 2. 1. 0.
TRI 1 1 2 3
TRI 2 3 2 4
TRI 3 3 4 5
TRI 4 5 4 6
TIME 87.000000
1.01      ! Flux for node 1 at T = 300
1.02      ! Flux for node 1 at T = 1000
2.01      ! Flux for node 2 at T = 300
2.02
3.01
3.02
4.01
4.02
5.01
5.02
6.01
6.02
TIME 90.000000
11.01
11.02
12.01
12.02
13.01
13.02
14.01
14.02
15.01
15.02
16.01
16.02
```

Figure 8. Example of BCM file format.

Once the formatted file has been created, the file can be used as input to the boundary condition mapper. After the file has been read into Thermal Desktop, the BCM will be presented as a mesh, as shown in Figure 9. The remaining mapping procedures are shown in Figures 10 through 12.

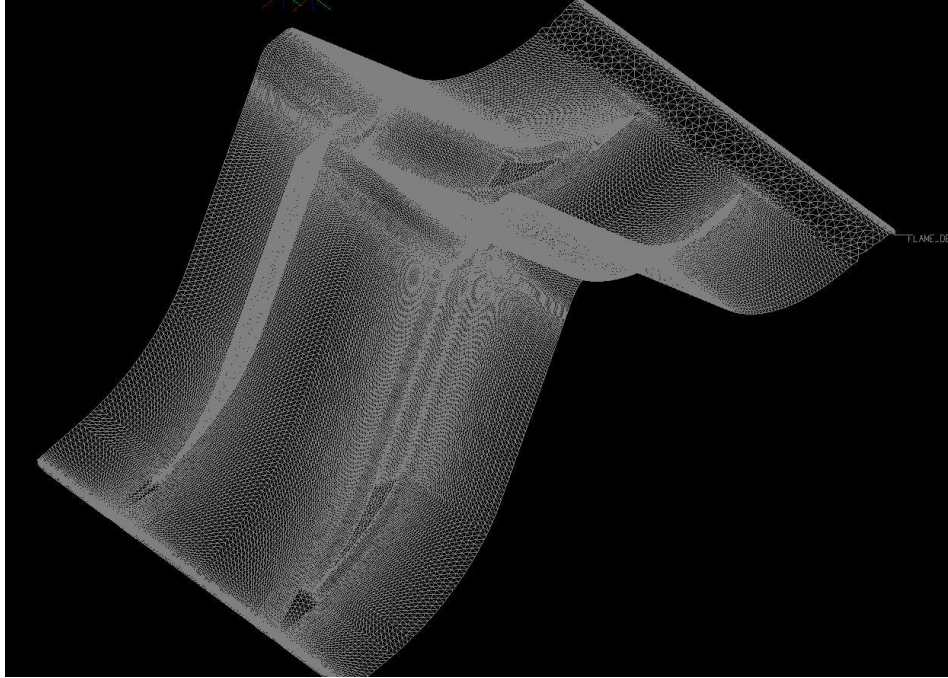


Figure 9. BCM mesh extracted from CFD model.

Using the AutoCAD align or move commands, the analyst must overlay the BCM on the thermal model surface to obtain an accurate mapping of data. If this is not done, the points will require high tolerances to map successfully, causing inaccuracies in the solution or causing points to fail to map.

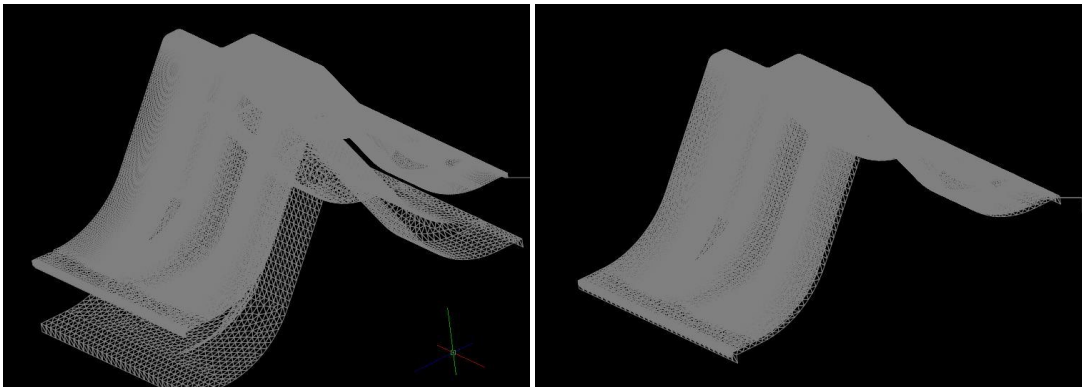


Figure 10. Aligning BCM to thermal model.

The BCM can be edited to point to the thermal model elements that are desirable for mapping the data onto and to specify variable tolerances. To successfully map all the points, a sufficient range of tolerances should be specified. In the case of this analysis, the option to apply a surface thickness to test points should be deselected. Deselecting this option prevents the test points for mapping to be generated based on the surface thickness. This is shown in Figure 11.

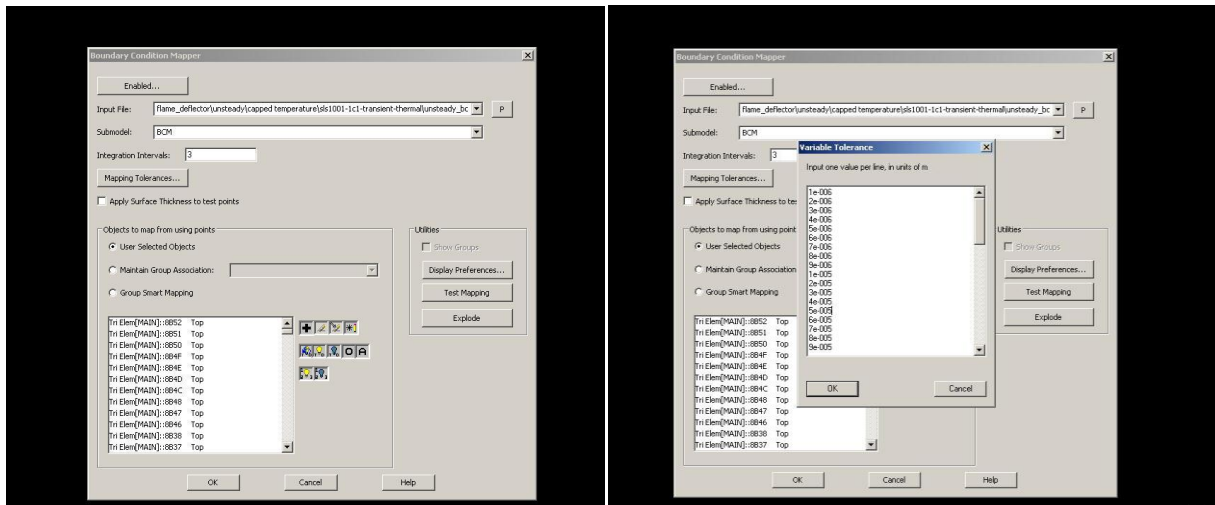


Figure 11. Boundary condition mapper setup window: (L) Selecting elements and (R) specifying mapping tolerances.

After selecting the thermal model nodes to be mapped and specifying tolerances, the analyst can use the test mapping tool to help ensure that all points have been successfully mapped before starting the simulation. Figure 12 is an example of a successful mapping.

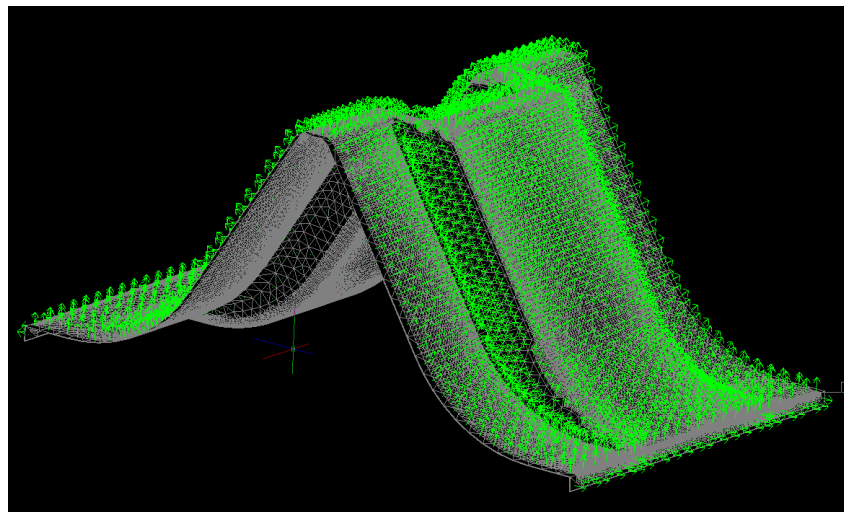


Figure 12. Successful mapping of the heat flux boundary.

POSTPROCESSING

The postprocessing of data in Thermal Desktop displays heat rate, heat flux, and temperature contours in a manner that the analyst can understand intuitively. However, in displaying contours of surface thicknesses, the postprocessing is not as straightforward. After completion of the processing, the ablation subroutine outputs a text file. The text file must then be

imported into the postprocessing datasets. These steps are illustrated in Figures 13 through 15. Figure 13 illustrates the Postprocessing Datasets window.

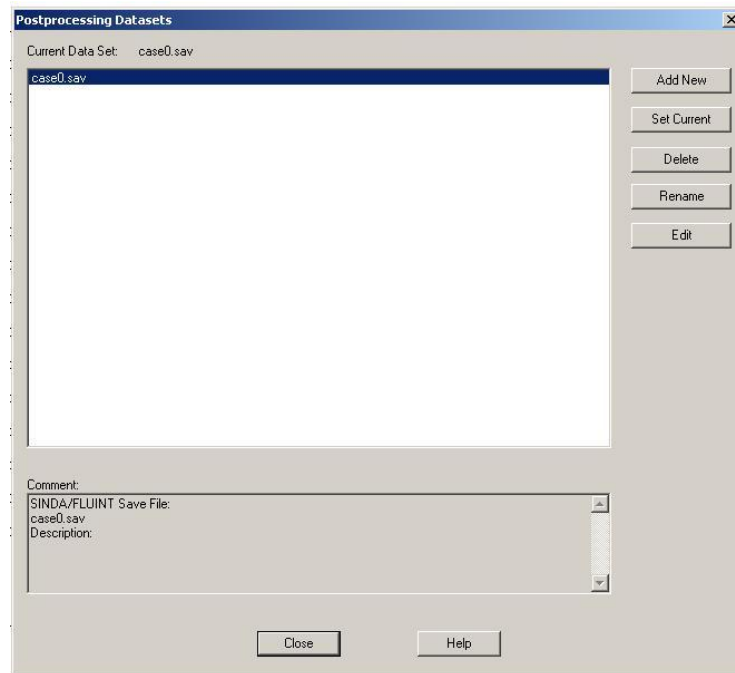


Figure 13. Postprocessing datasets import window.

The analyst can add new datasets that have color contours to the postprocessor. By selecting Add New and choosing a text transient file, the surface thickness time history text file can be imported into the postprocessor. Figure 14 illustrates the Data Set Source Selection window, as well as the drop-down menu used to select the file.

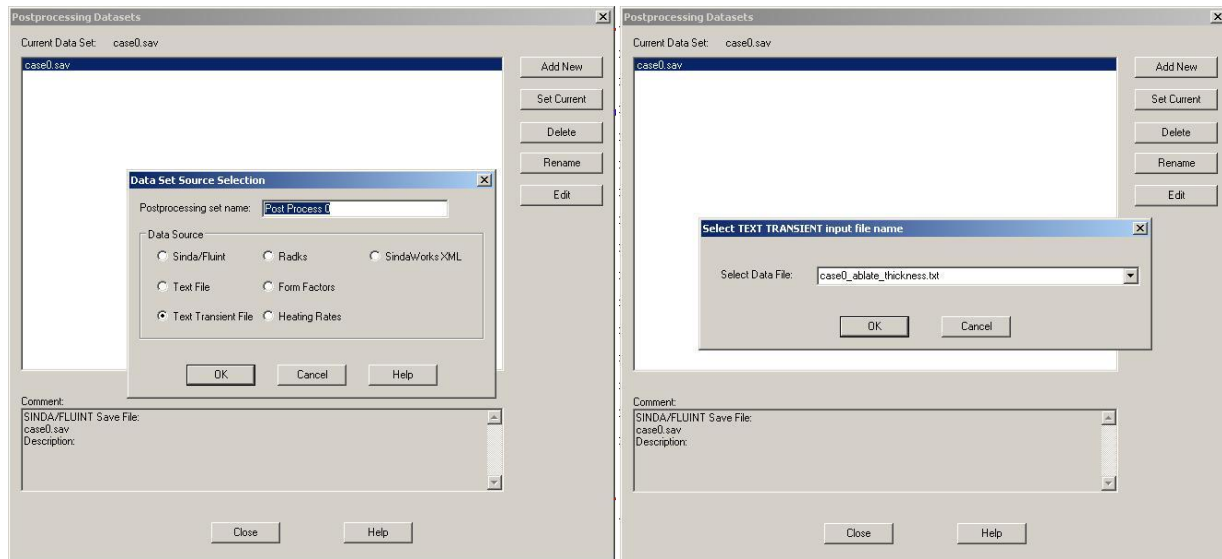


Figure 14. Text transient file import window.

Once the file has been chosen, the Set Transient Text Dataset Properties window will appear. Because of the existence of ablation nodes, and for any model using some form of insulation, the analyst should select Plot MLI from the drop-down menu. The data that exist on either the top or bottom sides will be plotted, depending on what the analyst selects. For the analysis of the flame deflector, “Plot MLI on top out sides” was chosen to capture the ablation nodes on the surface. Figure 15 shows the window used for setting the transient text dataset properties.

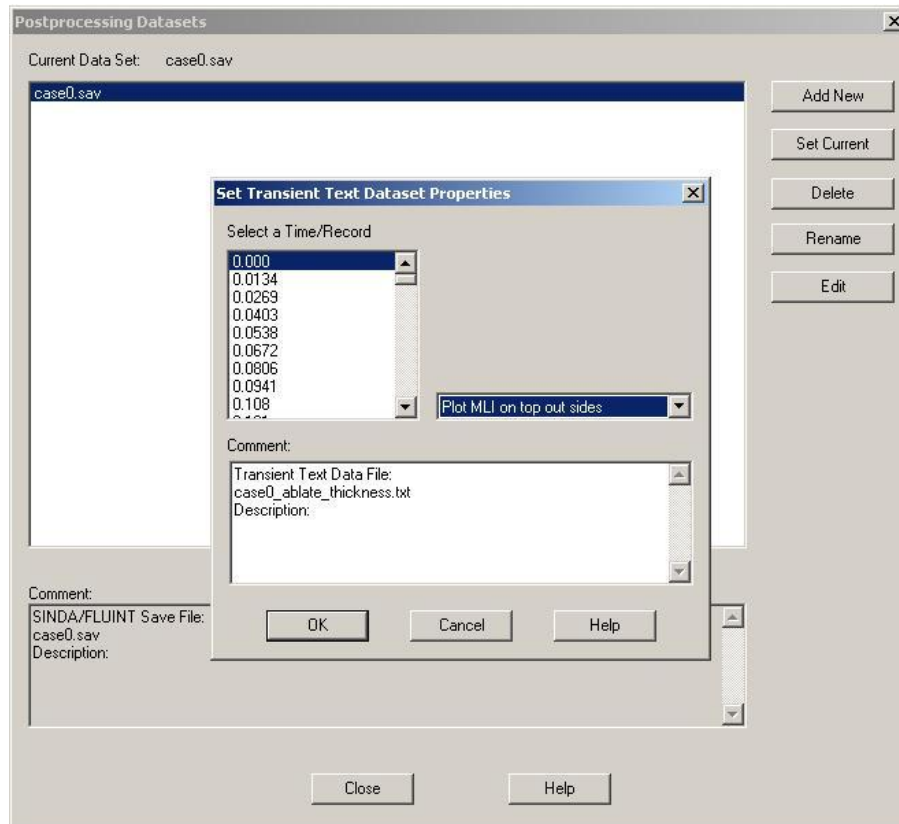


Figure 15. Set transient text dataset properties.

RESULTS

Heat Flux Data Mapping Comparison

The heat flux data used in the Thermal Desktop model is extracted from computational fluid dynamic (CFD) models provided by Ames Research Center. The heat flux data is computed from a conjugate heat transfer model where the maximum temperature is intentionally capped at the melting temperature of the refractory material. The melting temperature is approximately 1373 K. The mapping of the heat flux data shows good qualitative comparison between the CFD result and the Thermal Desktop result, as shown in Figure 16. The difference in heat flux magnitude is caused by the difference in surface area between the elements of the CFD model and those of the Thermal Desktop model. The heat flux magnitude determined by Thermal Desktop tends to be a lower value than the CFD data provided because of the significantly coarser mesh used for the thermal model.

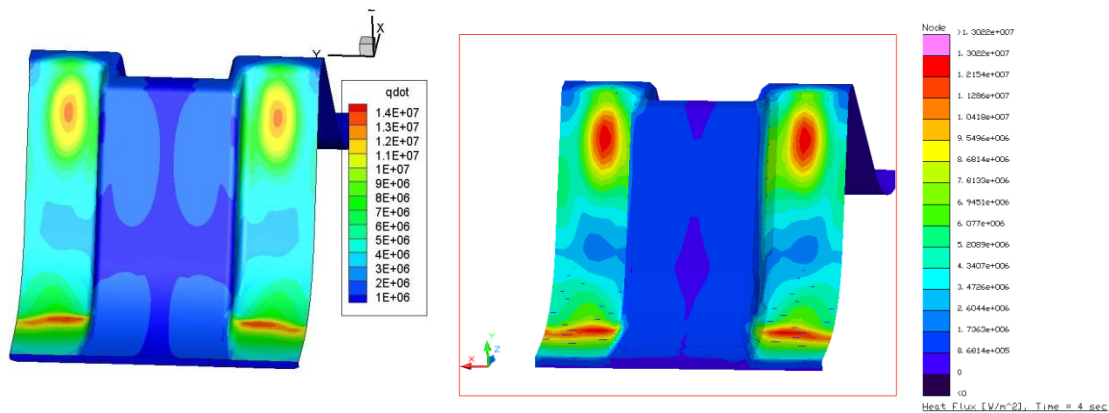


Figure 16. Comparison of heat flux data from CFD solution using Tecplot (L) and mapped data using Thermal Desktop model (R).

Surface Thickness Results

Using the Thermal Desktop analysis, the analyst can produce results for the mapped heat flux and surface thickness. The base material surface temperature contour can also be produced; but for this analysis, the temperature change of the base material was negligible and out of scope because the effort focused on the ablation and the capabilities of the ablation subroutine. Figures 17 through 20 and Table 1 summarize the results of the analysis.

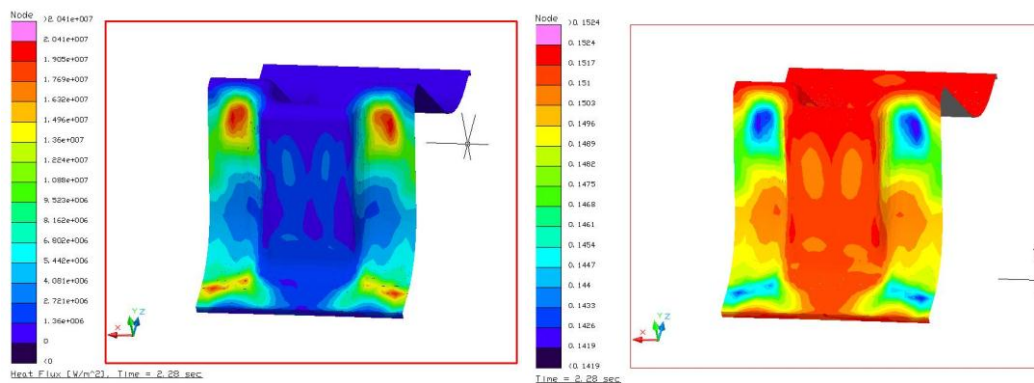


Figure 17. Heat flux and surface thickness contours for SLS simulation.

In Figure 17 the maximum heat flux and the maximum ablation are shown at the impingement locations in the upper corners of the flame deflector. The areas forming a secondary shockwave, shown in the lower corners of the flame deflector, also contribute a significant amount of heat flux and surface ablation. This is better depicted in Figure 18, which presents surface thickness versus time at the four locations.

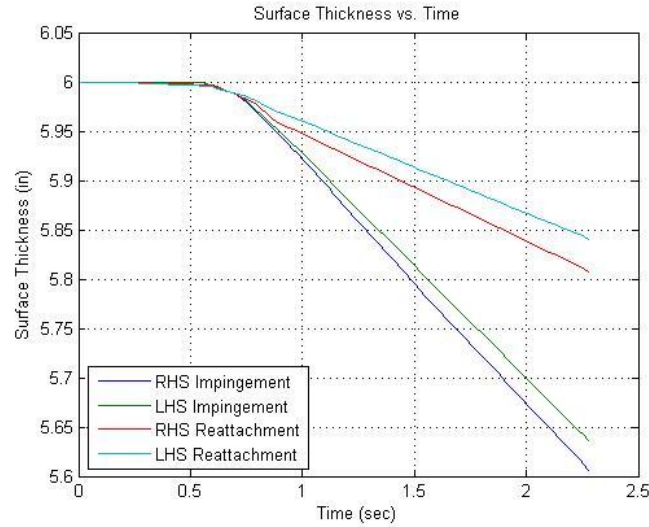


Figure 18. Surface thickness versus time at the impingement and reattachment regions.

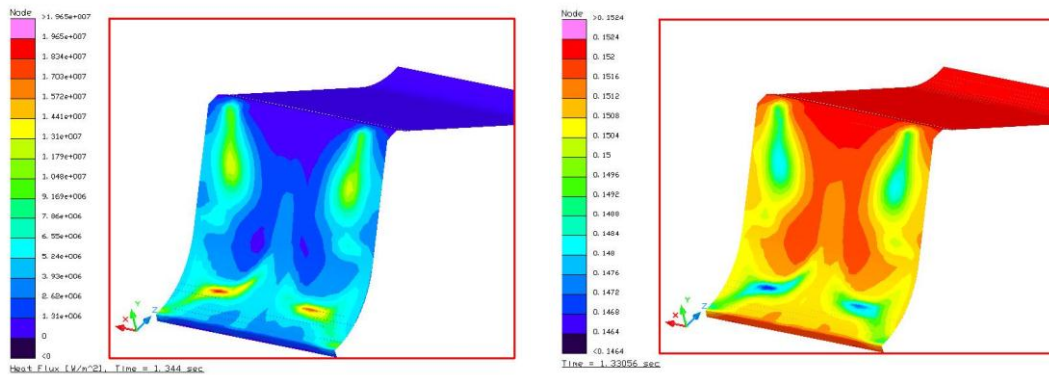


Figure 19. STS validation case.

The STS validation case was performed to evaluate how well the predictions agreed with experimental data for both the CFD analysis and thermal modeling. This analysis is still in progress, and we plan to refine the models to gain a better understanding in the future. The maximum loss of ablation thickness, which was measured after an STS mission, was approximately 0.229". From the STS validation thermal model, we determined the maximum ablation thickness to be 0.206" at full thrust conditions. Figure 20 shows the change in surface thickness versus time.

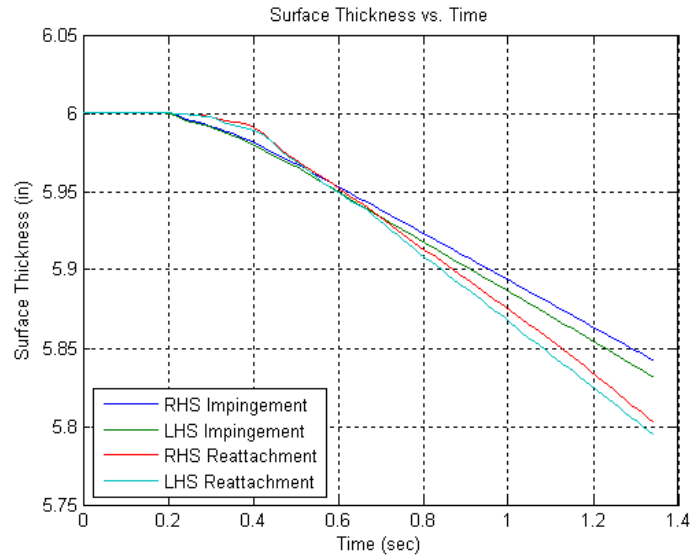


Figure 20. STS validation case surface thickness versus time.

Table 1. Summary of Results

Analysis	Time (sec)	Change in Surface Thickness (in)	Ablation Rate (in/s)
STS Validation	1.144	0.206	0.175
SLS Concept	1.710	0.394	0.230

CONCLUSIONS

Thus far the analysis has shown the benefits in using CFD and Thermal Desktop tools in tandem by employing the boundary condition mapper. The highly transient, highly compressible flow from the rocket plume can be modeled using high-fidelity numerical CFD codes. By extracting the thermal data from CFD models, one can perform thermal analyses that can better benefit future designs. In addition, the knowledge gained from using the ablation subroutine gives our analysis group a tool for ablation analysis in the future.

The flame deflector results were close to the expected 0.229" surface thickness loss measured after launch of an STS mission, while only simulating up to full thrust conditions. The increased material loss for the concept deflector for SLS is a result of the modeling of the five-segment solid rocket boosters, which produce significantly more heat than the four-segment boosters used in the STS program.

ACKNOWLEDGEMENTS

The authors would like to thank Cetin Kiris and Emre Sozer for their work in providing heat flux data from their CFD models for this Thermal Desktop analysis. This work was funded by the Mobile Launcher Launch-Induced Environment project, Task Order 51.

NOMENCLATURE, ACRONYMS, ABBREVIATIONS

ARC Ames Research Center

BCM boundary condition mapper

CFD computational fluid dynamics

KSC Kennedy Space Center

SLS Space Launch System

STS Space Transportation System

REFERENCES

Dimensional Evaluation of Pad A East Flame Trench Walls and Diverter Pre & Post STS-126 Launch, December 2008

SINDA/FLUINT User's Manual, C&R Technologies

Thermal Desktop User's Manual, C&R Technologies